Powder Technology 223 (2012) 39–45

Contents lists available at ScienceDirect

Powder Technology

journal homepage: www.elsevier.com/locate/powtec

Particle drying and crystallization characteristics in a low velocity concurrent pilot scale spray drying tower

M.W. Woo [a], S. Rogers [a], C. Selomulya [a], X.D. Chen [a,b,*]

[a] Department of Chemical Engineering, Monash University, Clayton Campus, VIC 3800, Australia
[b] Department of Chemical & Biochemical Engineering, College of Chemistry and Chemical Engineering, Xiamen University, Siming South Road, Xiamen, P.R. China

ARTICLE INFO

Available online 12 July 2011

Keywords:
CFD
Spray drying
Lactose
Mono-disperse
Crystallization
WLF

ABSTRACT

A low velocity spray tower was built in Monash which allows the production of mono-dispersed particles. As part of a larger project in controlling the functionality of the particles produced, the computational fluid dynamics technique (CFD) was used to gain more insights into the drying characteristics of the mono-dispersed droplets. Introduction of droplet and mass transfer did not significantly alter the flow field. Analysis revealed that the wet bulb region was significant in this tower. Varying the inlet air temperature from 100 °C to 180 °C resulted in contrasting drying history. These drying kinetics were then extended to assess the *in-situ* crystallization phenomenon. For this spray drying tower, it was found that lower inlet temperature condition favoured a higher degree of crystallinity.

1. Introduction

Spray drying technology has advanced from merely producing commodity particles to the current stage of controlling the functionality of the particles [1]. Such particle functionality control can be achieved by understanding and manipulating the drying history of the particle [2,3]. The drying history of a droplet is mainly affected by its trajectory in the hot drying air medium. Under relatively high air velocity flows, typical in spray dryers, trajectories of the droplets might not be uniformly controlled due to the intense turbulence dispersion. This poses a big challenge in the study of the droplet drying history and might impose an upper limit to the control of particle functionality.

For this purpose, a low velocity mono-dispersed spray tower was built in Monash. Unique feature of this dryer is in the usage of low air velocity; contrary to typical spray dryer configurations. To be more specific, from a previous numerical analysis, such low air velocity would have caused the mono-dispersed droplets to approach a free-falling trajectory pattern within the tower [4]. Nevertheless, even under such low velocity mild drying condition, this dryer has been successful in producing mono-dispersed particles. In our opinion, such low velocity spray drying concept can provide a new paradigm in controlling particle functionality by providing: (a) the possibility to induce mild drying conditions and (b) a more uniformly controlled particle drying history.

However, due to the fine droplet size used, experimental observation is mainly limited to the analysis of the product particles collected at the tower outlet which might not give sufficient details on the *in-situ* particle formation kinetics within the tower. Attempts were made to gain more understanding of the particle formation kinetics within the tower by undertaking one-dimensional simulations of the tower [5]. In this work, we will further extend the understanding of the system by visualizing the droplet drying behaviour within the tower using the Computational Fluid Dynamics (CFD) technique.

2. Geometry and simulation approach

2.1. Tower geometry and operation

Schematics of the spray tower are given in Fig. 1. Four air guns were placed on the ceiling of the tower. Total mass flow rate of air going into the tower was 1.45×10^{-2} kg s^{-1}. Below the inlets, a perforated distributor plate was installed to distribute the air. An outer annulus was also included in the design, so that an air curtain could be introduced along the wall during operation to minimize potential solid-wall impaction which could lead to deposition problems within the chamber. The bottom bustle comprises a flexible steel cone attached to the main tower. This tower is only partially insulated at the top half section which includes the ceiling wall.

2.2. Fluid flow and boundary condition settings

The simulation domain used is shown in Fig. 2. A total of 96,500 cells were used and mesh independence with this cell density was reported

* Corresponding author at: Department of Chemical & Biochemical Engineering, College of Chemistry and Chemical Engineering, Xiamen University, Siming South Road, Xiamen, P.R. China. Tel.: +86 592 2189911; fax: +86 592 2188855.
E-mail addresses: dong.chen@eng.monash.edu.au, xdc@xmu.edu.cn (X.D. Chen).

0032-5910/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.powtec.2011.06.030

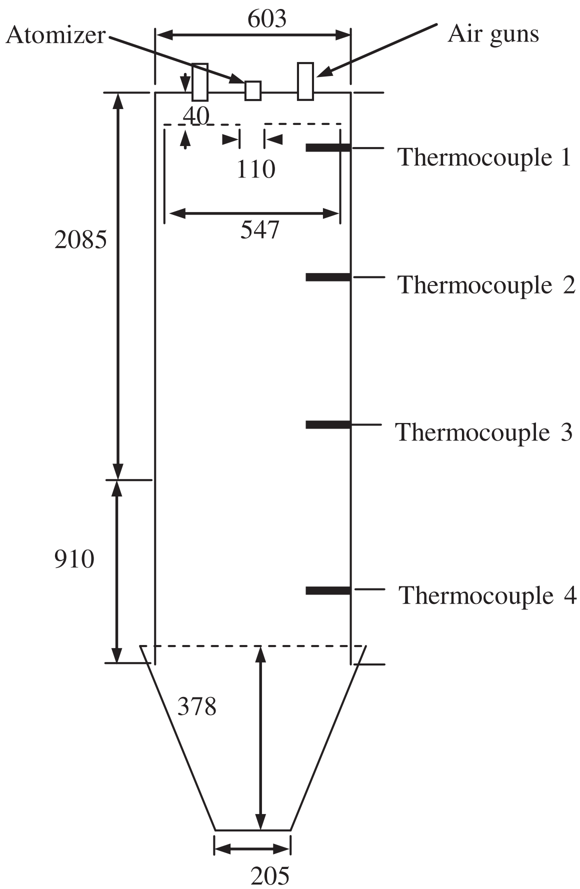

Fig. 1. Schematic of the drying tower (dimensions in mm).

in a previous work [4]. An axisymmetric model was used in the simulation, undertaken with FLUENT on the Ansys Workbench (version 12) platform. Details of the simulation were given in a previous report [4], but they are briefly repeated here for the benefit of the readers. As a simplifying assumption, the simulation domain did not include the ceiling region above the perforated plate and the air from the hot-air guns was assumed to be evenly distributed circumferentially in the plenum region above the distributor plate. The simulation domain reported here follows that of Case H in Woo et al. [4].

In line with the axisymmetric assumption, the perforated plate was modeled as an annulus inlet with uniform velocity distribution, while the central region was modeled as a separate circular inlet in the centre. The outer annulus was modeled as a uniform annulus inlet. Proportion of the air entering from each annulus boundary was distributed proportionally to the actual physical open surface area of the boundary [4]. It is noteworthy that the mass flow rate used at the center open surface area also included 3.13×10^{-4} kg s^{-1} of the dispersion air near the atomizer.

Experimental measurements showed that, particularly at high temperatures, significant heat loss occurs at the plenum chamber; which resulted in a lower air temperature within the plenum chamber (Fig. 3). As the plenum chamber was not included in the simulation, the inlet air temperature into the simulation should reflect that of the plenum chamber. In subsequent discussion, the term 'inlet temperature' denotes the temperature setting of the air guns. However, the actual air temperature setting for the simulation was based on the corresponding plenum chamber air temperature as shown in Fig. 3.

The Realizable k-ε model was used based on a previous report for similar confined jet system where a central jet interacts with an outer annulus jet [6]. The inlet water vapour mass fraction at the inlet corresponded to an ambient condition of 60% RH at 24 °C as extracted by the hot air guns. The degree of turbulence at the inlet was assumed to be mild exhibiting 10% turbulence intensity and the dissipation length scale corresponded to the characteristic lengths of the inlet geometries [7]. In view of the partial insulation of the chamber, the boundary heat loss coefficient was chosen *a priori* by making comparison with dry runs of the tower [8]. The measurement and comparison of the axial air temperature at different tower heights have been reported previously [4]. Table 1 summarizes the boundary conditions used. The term 'backflow' in Table 1 refers to the flow going into the simulation domain in the event that the pressure within the chamber is lower than the external ambient condition. This phenomenon was observed in the gap between the cylindrical tower body and the bottom conical geometry.

Transient simulation was undertaken and a time step of 0.005 s was used for the final flow field analyzed in each case. At each time step, the final scaled residuals were ensured to be lower than 0.0001. A transient air-steady particle injection approach was used to post-process the particle drying history [4,7,8]. This method does not necessitate transient sampling of the particle which requires excessive computational time and the effectiveness was previously shown for this system [4]. However, for this post-processing method to reflect on the actual drying condition, it will be important to fully develop the humidity and temperature field (on top of the momentum flow field) prior to post-processing. The flow field was firstly developed by monitoring and achieving a stable axial velocity at the bottom outlet. Transient particle injection was then started and the humidity and temperature field development was checked by monitoring the humidity and temperature at the outlet. When discrete particles were injected into the developed flow field, heat and mass transfer from the droplet would decrease the air temperature while at the same time increase the air humidity. This would be reflected in changes of the humidity and temperature of the outlet air. When these parameters at the outlet reached an apparent stable condition, the humidity and temperature within the chamber were assumed to be fully developed. Further analysis was therefore only undertaken with the developed flow field.

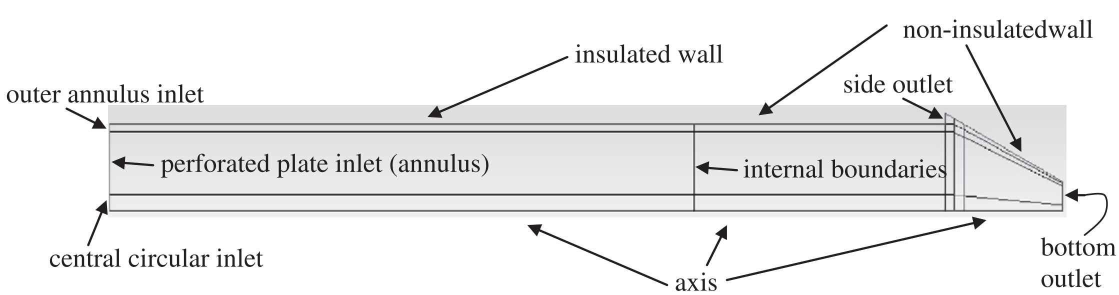

Fig. 2. Simulation domain implemented in the model.